Polytechnic University of Catalonia

Master in Numerical Methods / Master on Computational Mechanics

Communication Skills I
Assignment 2: Extended Abstract

Bruno A. Tessaro and Lisandro A. Roldan

January 25, 2016

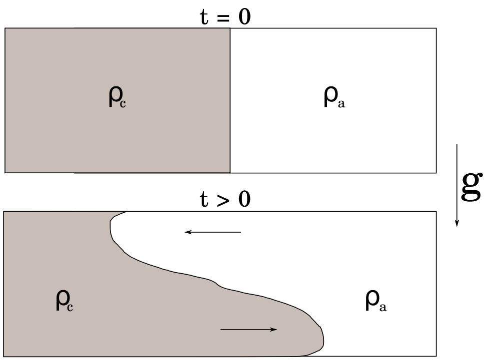
Project

Direct Numerical Simulation of Particle-laden gravity currents using OpenFOAM by Bruno Aguirre Tessaro, Jorge Hugo Silvestrini.

Keywords: Gravity Currents, Turbulence, CFD, OpenFOAM.

1 Introduction

Gravity current is a physical phenomenon that occurs when two bodies of fluids with different densities meet. The heavier fluid will flow under the lighter fluid generating a current as shown in Figure 1.1a There are abundant examples of this events in nature, such as sandstorms (Figure 1.1b), avalanches, sea floor turbidity currents, among others.



(a) Schematic of gravity currents

(b) Sandstorm in Kuwait

Figure 1.1: Gravity currents.

They can be classified in conservatives or non-conservatives depending on the formation mechanism. For the first one, flows are a product of differences of temperature or salinity, while in the other they are produced by the presence of solid particles in suspension. This study focuses on this last type of events.

It's relevance resides in the proper design of submarine equipment and structures as well as the understanding of the formation process of landscapes and petroleum reservoirs.

2 Theoretical Reference

This phenomenon is mathematically modelled with the use of some assumptions to make it computationally affordable.

- Disperse phase sufficiently diluted.
- Negligible viscosity variations.
- Incompressible flow.
- Negligible inertial interactions between particles.
- Solid particles with uniform size (mono-disperse) and spherical shapes.
- Negligible erosion or re-suspension.
- Particles do not accumulate on the bottom off the domain, but are computed as post-processing.

The differential equations that govern the behaviour of the fluids are: conservation of mass, conservation of momentum (with Bussinesq approximation) and a concentration transport equation, taking into account a reduced gravity term and the strokes law of sedimentation, as shown as:

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{2.1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho_a}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + c\tilde{g}e_i^g, \tag{2.2}$$

$$\tilde{g} = \frac{C_r(\rho_p - \rho_a)g}{\rho_a}, \tag{2.3}$$

$$\frac{\partial c}{\partial t} + (u_j + u_s e_g^j)\frac{\partial c}{\partial x_j} = \kappa \frac{\partial^2 c}{\partial x_j^2}, \tag{2.4}$$

$$u_s = \frac{d_p^2(\rho_p - \rho_a)g}{18\mu}. \tag{2.5}$$

3 Methodology

The implementation of a new solver is done in OpenFOAM, a CFD toolbox, because of its capability of working with complex geometries. It has also the advantages of being an open source platform, it is widely used in industry and academic research and possess a wide library of pre-programmed solvers.
The implementation consists in taking the base solver *twoLiquidsMixingFoam* , which solves a similar problem (mixing of two incompressible fluids), find the governing equations and replace them with the ones that hold for particle-laden gravity currents.
In order to validate this new solver, an already published and validated work is used as reference [1]. This work solves the problem of particle-laden gravity currents using a high order finite difference code called Incompact3D [2], developed in the Imperial College of London. Numerical and physical parameters being used in this work are the same as the reference work.

The physical configuration used is the well known "lock-exchange" case, which consists in a squared domain with a gate separating two fluids with different densities, as shown in Figure 3.1. Once the gate is opened, the phenomenon occurs.

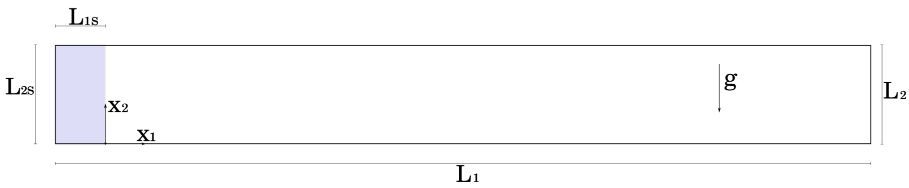
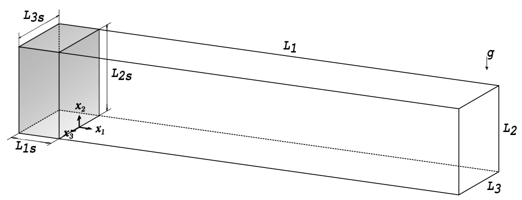
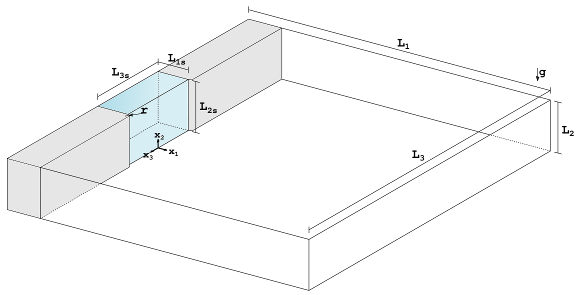


Figure 3.1: Lock-exchange configuration, shaded area represents the diluted phase.

Post processing is carried out with the same code used by the reference work to further compare the results obtained. Position of the front of the current, deposit profile and energy balance are calculated.
Two more simulations are carried out with a course mesh in order to show the capability of OpenFOAM to work with different geometries, they can be seen in Figure 3.2.



(a) Lock-exchage 3D



(b) Channel-Basin 3D

Figure 3.2: Test cases

4 Results

In Figure 4.1, a comparison between the concentratio field results from the different codes can be observed. In the initial times, the shape of the current are very similar, varying as the time passes. This behaviour is expected due to the fact that this is a dynamical system and that OpenFOAM was not designed for DNS simulations.

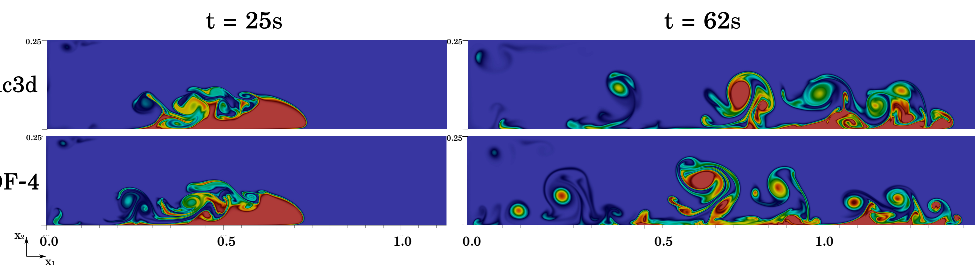
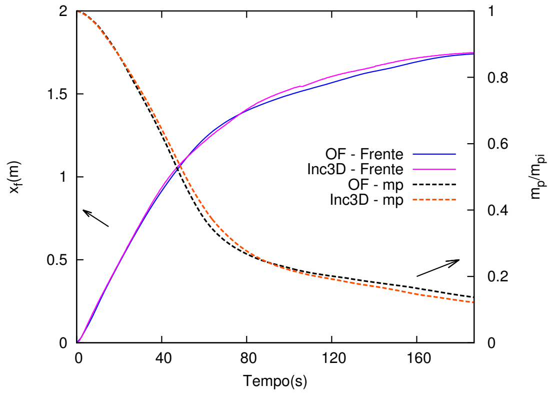
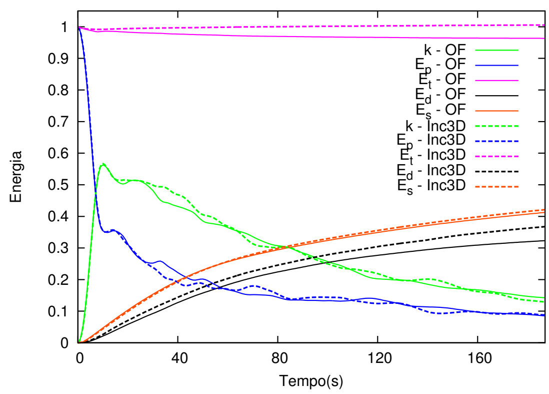


Figure 4.1: Concentration field comparison for different times, on top, reference results and on bottom, OpenFOAM results.

Furthermore, the post processing results are presented in Figures 4.2. Figure (a) represents the position of the front of the current in time as well as the amount of suspend particles in time. Figure (b) shows a energy busdget comaparison, in the initial time, all the energy is potential energy (E_p), which will be converted in kinect energy (k) and its going to be dissipated due to the presence of particles (E_s) and turbulence (E_d). The most evident difference can be spotted in the turbulent dissipation, which reflects on the total energy (E_t). Again, this behavior can be explained due to the fact that OpenFOAM is not made to deal with turbulence.



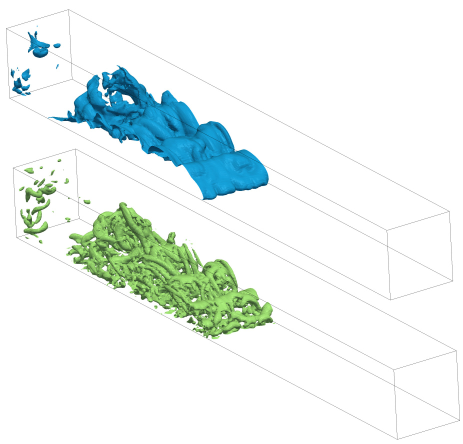
(a) Front position and suspended mass



(b) Energy budget

Figure 4.2: Graphical representation of the post processing

The 3D cases results are shown in Figure 4.3. It can be observed that even with a coarse mesh the physics of the problem are preserved and the results are coherent.


(a) Front position and suspended mass

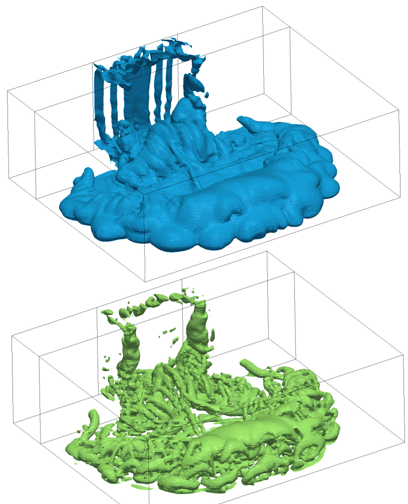

(b) Energy budget

Figure 4.3: Graphical representation of the post processing

5 Conclusion

It can be concluded that the results obtained by the new solver show accuracy in comparison to the reference work. Also, good results were obtained for complex geometries in the test cases, considering that they were carried out with coarse meshes. Now validated, the new solver can be used to further research in the gravity currents area field.

References

[1] Espath, L., Pinto, and Silvestrini, J. Two-and three-dimensional direct numerical simulation of particle-laden gravity currents. *Computers & Geosciences 63* (2014), 9–16.

[2] Laizet, S., and Lamballais, E. High-order compact schemes for incompressible flows: A simple and efficient method with quasi-spectral accuracy. *Journal of Computational Physics 228*, 16 (2009), 5989–6015.